Gerber file generation for PCAD 2001

David Wentzloff
david@wentzloff.com
781-322-7769

Main Tasks Explained in This Document:

A. Finalize PCB design and check it
B. Generate GERBER files
C. Generate N/C Drill files
D. Verify the GERBER files look correct

A. Finalize PCB design and check it

1. Run DRC (Utils > DRC…) and compare the PCB netlist with the schematic netlist
2. Pour all copper pours
3. Silkscreen line width is recommended to be at least 7mils

B. Generate GERBER files

4. Add text at the top left of the design, outside the board perimeter, that says the layer name in the layer (i.e. type top copper in the 'top' layer, bottom copper in the 'bottom' layer, …). Name the following layers for a 2 layer board
   - 4.1. Top copper
   - 4.2. Top silk
   - 4.3. Top solder mask
   - 4.4. Bottom copper
   - 4.5. Bottom silk
   - 4.6. Bottom solder mask
5. Bring up the File Gerber Out menu (File > Export > Gerber…), then click Setup Output Files…
6. From the Setup Output Files dialog, add each of the following layers and options

6.1.

Layers:	Top, Board
File Extension:	TOP
Options:	Pads, Vias

6.2.

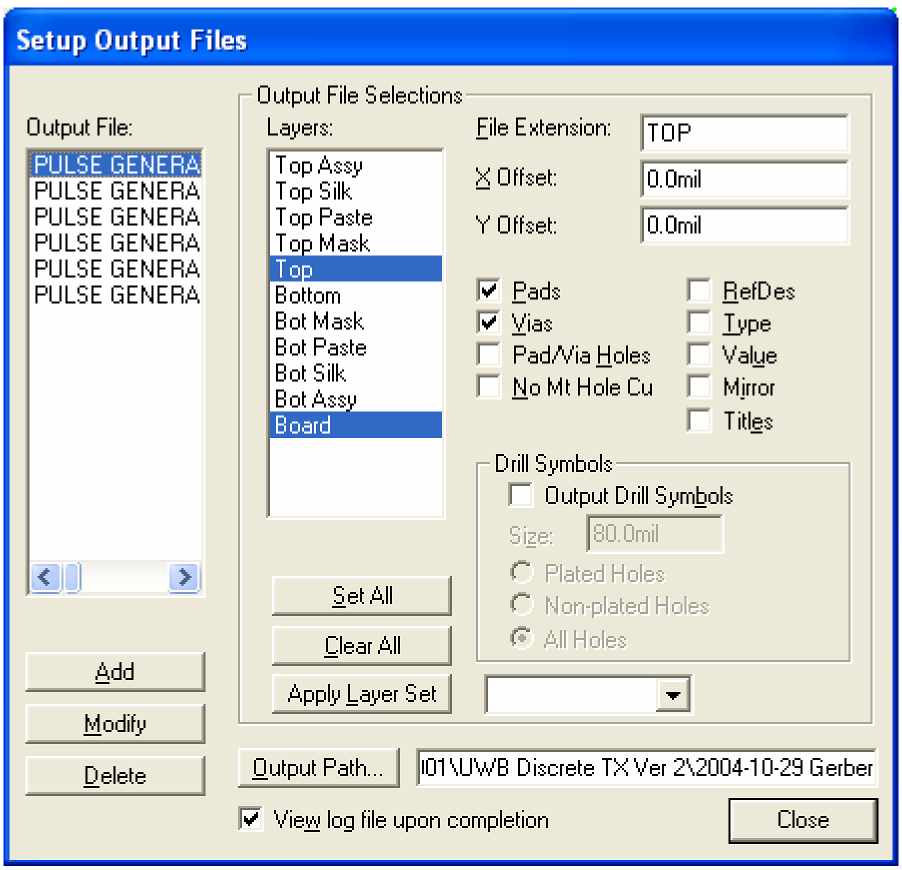
Layers:	Bottom, Board
File Extension:	BOT
Options:	Pads, Vias

6.3.
Layers: Top Silk, Board
File Extension: TSK
Options: RefDes, Type, Value

6.4.
Layers: Bottom Silk, Board
File Extension: BSK
Options: RefDes, Type, Value

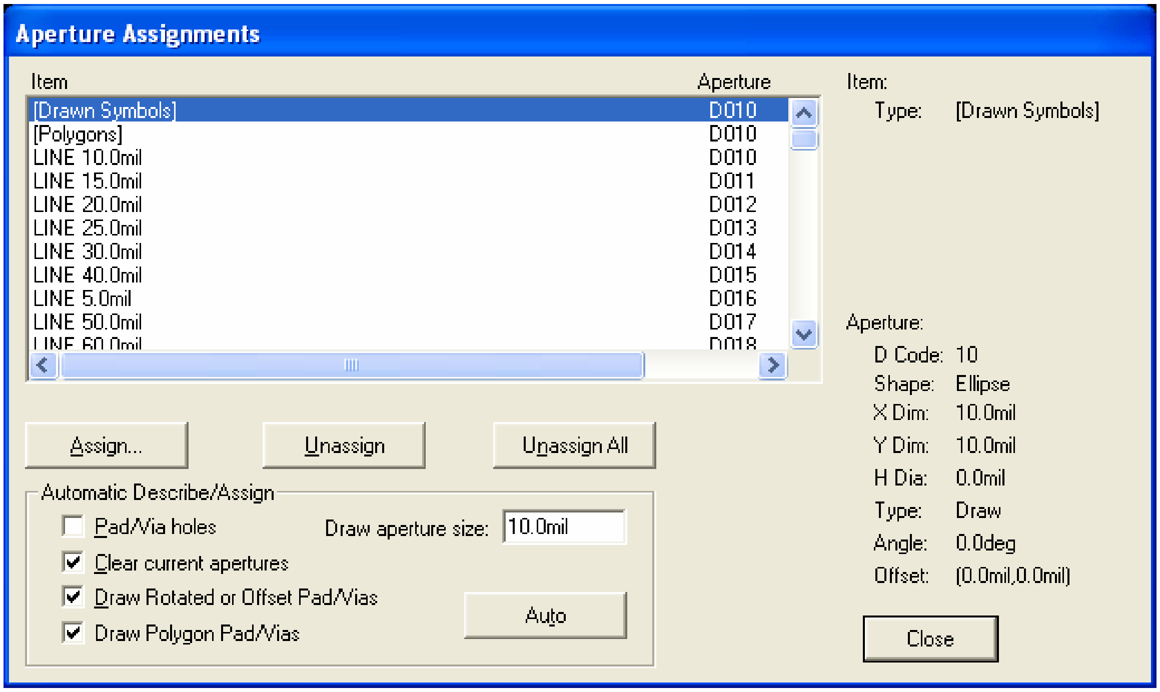
6.5.
Layers: Top Mask, Board
File Extension: TMK
Options: Pads, Vias, Pad/Via Holes

6.6.
Layers: Bottom Mask, Board
File Extension: BMK
Options: Pads, Vias, Pad/Via Holes

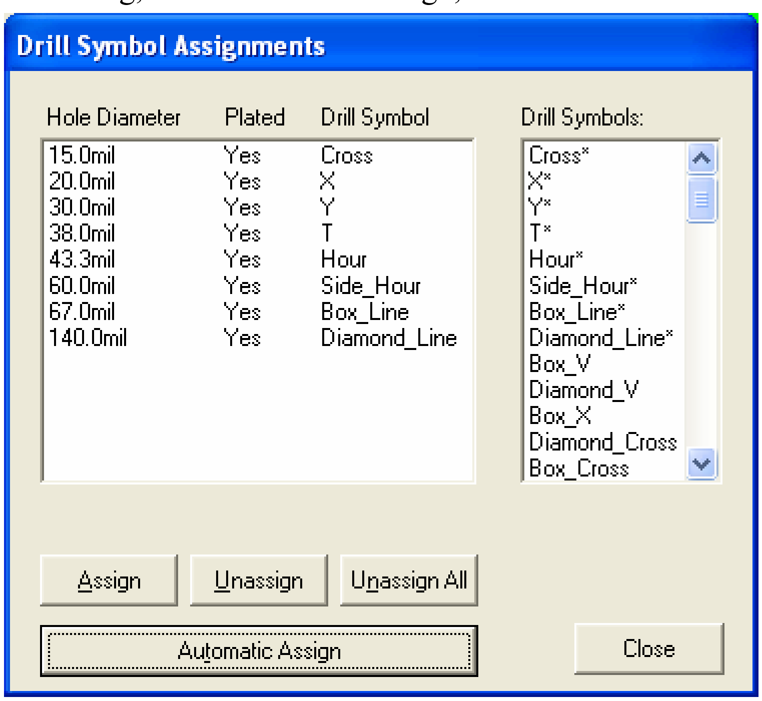


7. Click Close in the Setup Output Files dialog, then click Apertures… in the File Gerber Out dialog

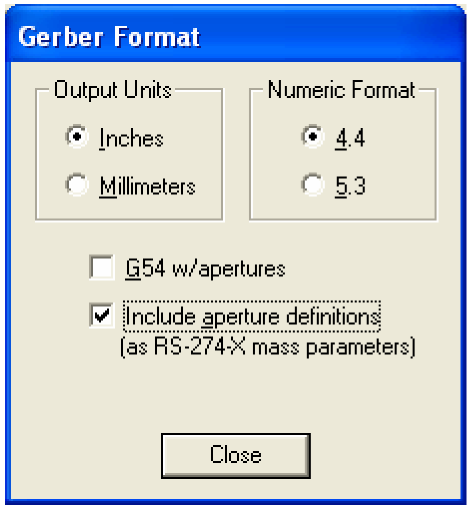
   7.1. Verify Pad/Via holes is unchecked, and the other options are all checked, then click Auto and Close



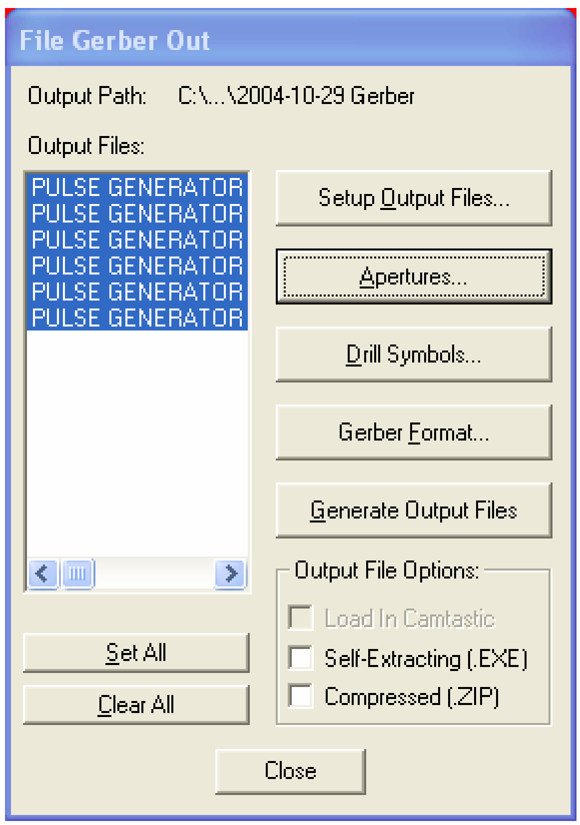
8. From the File Gerber Out dialog, click Drill Symbols… In the Drill Symbol Assignments dialog, click Automatic Assign, then Close



9. From the File Gerber Out dialog, click Gerber Format… Select Inches, 4.4 format, and check Include aperture definitions (as RS-274-X mass parameters). Click Close



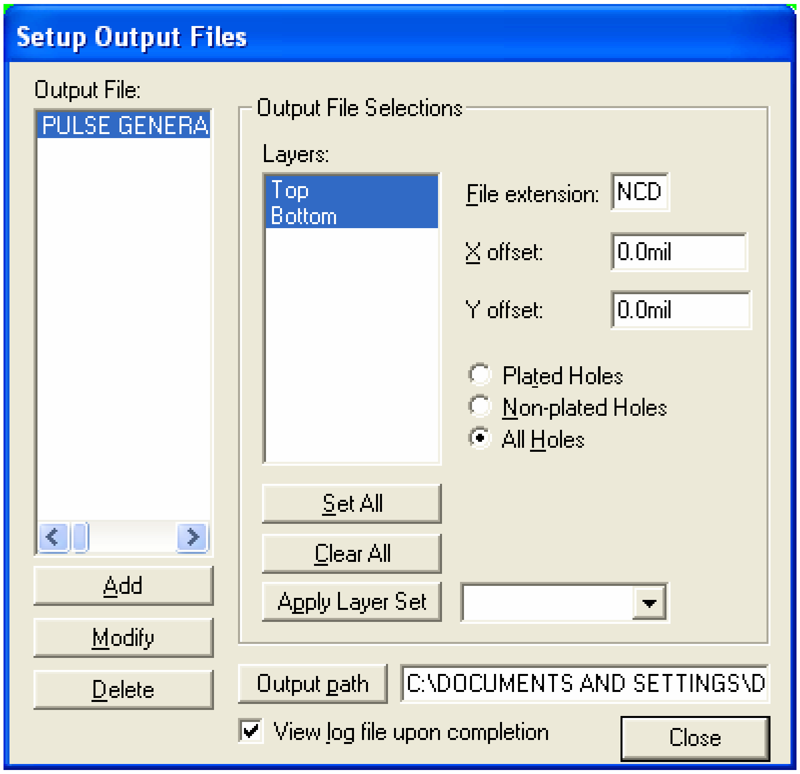
10. The final File Gerber Out dialog should look like the following. Click Generate Output Files and then Close.



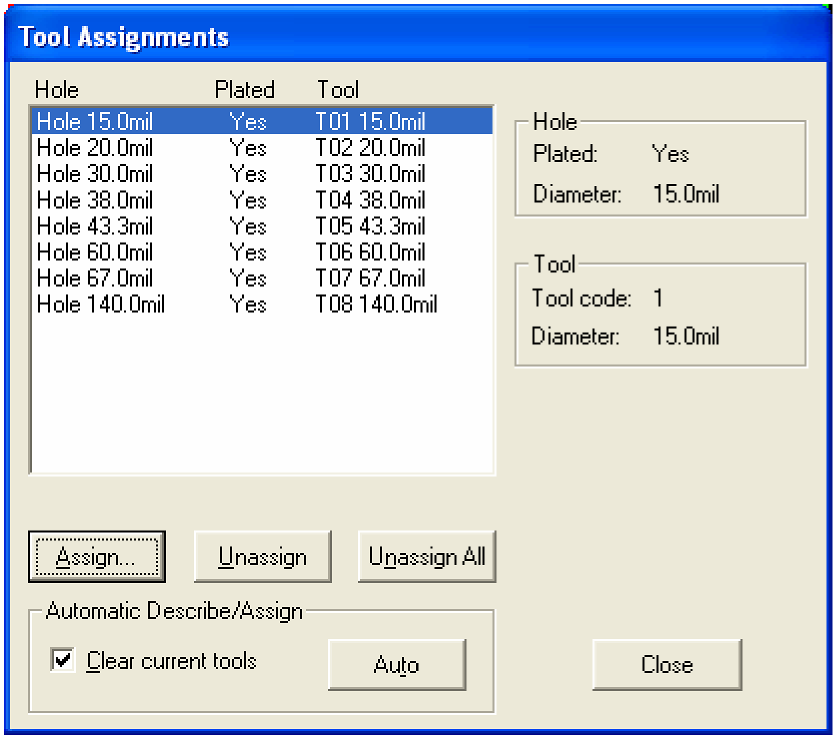
C. Generate N/C Drill files

11. Select File > Export > N/C Drill File to bring up the File N/C Drill dialog. Click Setup Output Files…

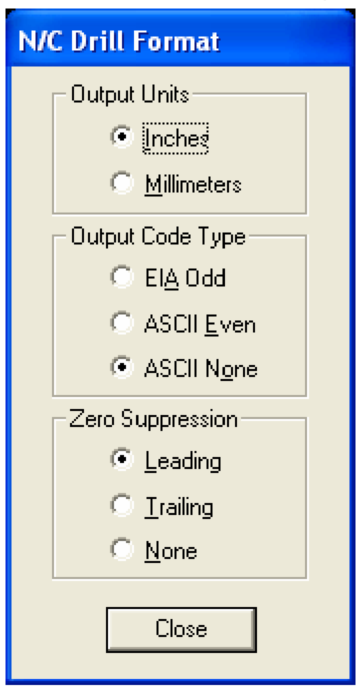
    11.1. Select all signal and plane layers, file extension should be NCD, check All Holes, and verify the Output Path is correct. Then click Add and Close.



    11.2. Click Tools, then Unassign All, Auto, and Close



11.3. For the N/C Drill Format, use the following settings



11.4. Click Generate Output Files and Close

D. Verify the GERBER files look correct

12. Verify you design by importing the GERBER and N/C Drill files into a free GERBER viewer such as GC Prevue available from http://www.graphicode.com/